Tech Memo
P 1205

Tech Memo
P 1205

AD-A241 597

DEFENCE
RESEARCH
AGENCY

AEROSPACE DIVISION

DTIC
ELECTE

Technical Memorandum

April 1991

Predictions and Measurements of 3D Viscous Flow in a Transonic Turbine Nozzle Guide Vane Row
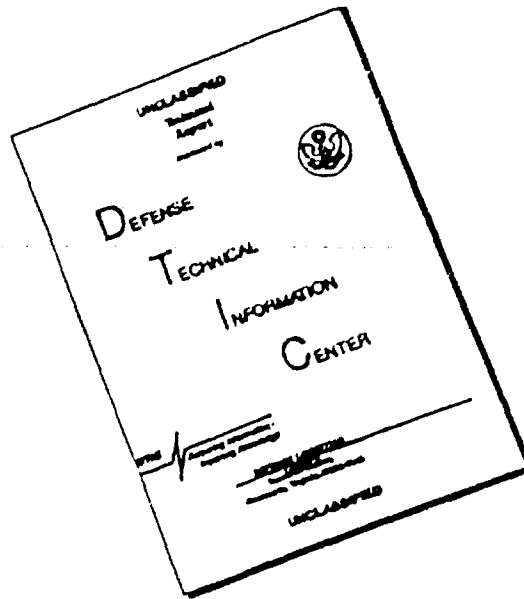
by

G. C. Horton
S. P. Harasgama
K. S. Chana

91-12271

RAE, Farnborough, Hampshire

DISCLAIMER NOTICE

THIS DOCUMENT IS BEST QUALITY AVAILABLE. THE COPY FURNISHED TO DTIC CONTAINED A SIGNIFICANT NUMBER OF PAGES WHICH DO NOT REPRODUCE LEGIBLY.

CONDITIONS OF RELEASE

1108272   303827

MR PAUL A POSEY
STIC
Attn:DTIC-FDAC
Cameron Station-Bldg 5
Alexandria
VA 22304 5145
USA

....   DRIC U

COPYRIGHT (c)
1988
CONTROLLER
HMSO LONDON

....   DRIC Y

Reports quoted are not necessarily available to members of the public or to commercial
organisations.

DEFENCE RESEARCH AGENCY

Aerospace Division

RAE Farnborough

Technical Memorandum P 1205

Received for printing 25 April 1991

PREDICTIONS AND MEASUREMENTS OF 3D VISCOUS FLOW IN A

A TRANSONIC TURBINE NOZZLE GUIDE VANE ROW

by

G. C. Horton

S. P. Harasgama

K. S. Chana

SUMMARY

In recent years improvements in algorithms and computing power have allowed the regular use of three-dimensional viscous flow programs to analyse the flows in turbines. Before these programs can be used with confidence by the turbine designer, however, they must be validated by comparison with high quality experimental data taken at realistic conditions.

A transonic turbine nozzle guide vane has been tested in an annular cascade with two different endwall geometries. The measurements were taken at engine-representative flow conditions and include surface static pressures and a downstream area traverse of total pressure.

The flow through these geometries has been modelled at the test conditions using a three-dimensional viscous flow program. The effects of different mesh densities and two turbulence models have been studied. Predictions of secondary flow and loss have been obtained and are compared with the experimental measurements.

2

LIST OF CONTENTS

C.V. Hinton, S.P. Harasgama, K.S. Chana
Royal Aerospace Establishment, Pyestock
Farnborough, Hampshire
U.K.

SUMMARY

In recent years improvements in algorithms and computing power have allowed the regular use of three-dimensional viscous flow programs to analyse the flow in turbines. Before these programs can be used with confidence by the turbine designer, however, they must be validated by comparison with high quality experimental data taken at realistic conditions.

A transonic turbine nozzle guide vane has been tested in an annular cascade with two different endwall geometries. The measurements were taken at engine-representative flow conditions and include surface static pressures and a downstream area traverse of total pressure.

The flow through these geometries has been modelled at the test conditions using a three-dimensional viscous flow program. The effects of different mesh densities and two turbulence models have been studied. Predictions of secondary flow and loss have been obtained and are compared with the experimental measurements.

1.   INTRODUCTION

The use of three-dimensional viscous flow programs to analyse the flow in turbomachinery blade passages has become widespread in recent years. In general these methods are currently used to analyse existing designs or to check specific features of a new design in the final stages of the design process. If these methods are to realise their full potential for radically improving turbomachinery design they must be used much earlier in the design process. This requires the designer to have confidence in the capability of the program and also a good understanding of the effect of the available options, such as mesh size and density, or different turbulence models.

To help prove this confidence, a turbine nozzle guide vane (ngv), designed for transonic flows, has been tested in annular cascade form with two different endwall shapes in the Isentropic Light Piston Cascade (ILPC) facility at RAE Pyestock. The same nozzle geometry has been analysed using a three-dimensional viscous flow program with different meshes and turbulence models. This paper reports the results of these computations and makes comparisons with the experimental results.

2.   THE NGV DESIGN

The ngv tested was designed at the Royal Aerospace Establishment as part of a high work capacity, high blade speed turbine known as the High Rim Speed Turbine. The design philosophy and test performance have been

described in a previous AGARD paper (Ref 1) and blading aerodynamic and heat transfer measurements in Ref 2. The blade design is illustrated in figure 1. The design features a variation of exit flow angle along the span and a curved tangential stack of the trailing edge to control secondary flows by setting up spanwise pressure gradients. At the design condition the exit Mach number is 1.05 and the Reynolds number (based on exit conditions and axial chord) is 1.73×10^6. The original ngv was designed for cylindrical endwalls, but as part of an investigation of potential improvements to the turbine efficiency, profiled hub shapes were also considered.

The blading design has been tested with two of these endwall shapes, which are here termed the bellmouth and S-bend profiles (see figure 1). The aim of these endwall shapes was to reduce the secondary effects by re-energising the flow near the endwall. The distribution of curvature, and hence the local acceleration provided to the flow, varied between the two designs.

3.   THE ILPC FACILITY

The Isentropic Light Piston Cascade, or ILPC, (Ref 3) is a short duration facility designed to allow high quality heat transfer and aerodynamic measurements to be taken for a full-size annular cascade of turbine vanes. Its design and operation has been described in an AGARD paper presented in 1985 (Ref 3). A major extension of the facility, to enable heat transfer data to be taken from a rotating rotor mounted downstream of the nozzle row, has been designed and will be installed in 1991. The aerodynamic conditions upstream of the cascade, to match the engine parameters of exit Mach number, Reynolds number and gas-to-wall temperature ratio, are obtained by the isentropic compression of air in a large pump tube. This compression is performed by a free piston driven along the tube by high pressure air. When the correct conditions are achieved, a fast-acting plug valve opens allowing the air to flow through the cascade giving steady operating conditions for approximately 0.5 second, during which time aerodynamic and heat transfer data can be aquired.

For this series of tests the instrumentation consisted of static pressure tappings at a total of 188 locations on the vane surfaces and endwalls. An area traverse of total pressure was performed on a plane 0.2 axial chord downstream of the trailing edge. This was built up from a series of 27 circumferential sweeps at different radii, each sweep requiring a separate run of the ILPC facility. The circumferential extent of a sweep was approximately 1.5 vane pitches. As it was desired to traverse through two vane wakes on each sweep and the vane wakes were

significantly curved, regular adjustment of
the location of the traverse mechanism was
necessary to produce a full-area traverse.
As a result, producing the area traverse took
a considerable time and so was done for only
one endwall profile, the bellmouth profile.
Results reported previously for the cylindri-
cal endwall geometry (Ref 4) were obtained
with a three-hole probe with high-response
transducers mounted near the tip. These
results suffered from a high degree of noise,
much of it aerodynamic. For the current
series of traverses a modified probe configu-
ration was used. This configuration had
three transducers with lower frequency
response mounted 500 mm from the probe head.
This distance between the probe head and the
transducers appears to damp out the aerody-
namic noise effectively while providing
enough response to capture adequately the
vane wakes. The natural frequency of the
system is now approximately 400 Hz.

The manner of operation of the traverse
mechanism presented some difficulties when
measuring near the hub. To overcome these
difficulties two different probe tips were
used: a straight tip for the traverses above
mid-height and a "swan-necked" tip for those
below (figure 2). By using these two probe
tips it was possible to traverse to within
7.5% vane height of both endwalls.

4.   ANALYSIS METHOD

The method used to analyse the flow is the
three-dimensional viscous flow program of
Dawes (Ref 5). This solves the unsteady
Reynolds averaged Navier Stokes equations for
a steady solution using an implicit time
marching algorithm. Turbulent stresses are
modelled using an eddy viscosity concept.
Two formulations of eddy viscosity are
available: an algebraic method due to Baldwin
and Lomax (Ref 6) and a one-equation method
patterned after that of Birch (Ref 7). The
latter model includes an additional transport
equation for turbulent kinetic energy which
is solved using a space-marching algorithm.
In the Baldwin-Lomax model laminar-turbulent
transition is modelled by specifying loca-
tions for the start and end of transition on
both suction and pressure surfaces. A value
of intermittency is then calculated for each
mesh point by linear interpolation between
these specified transition locations. For
the current series of calculations tran-
sition on the suction surface was set to
start at 25% axial chord (Cax) and end at 60%
Cax. The corresponding figures for the
pressure surface were 40% and 90%. The
one-equation method models transition through
a low Reynolds number damping term.

Spatial discretisation is achieved using a
sheared, cell-centred H-mesh. Two different
densities of mesh were employed; calculations
with meshes 1 and 2 used 25 points circum-
ferentially, 25 radially and 89 axially, a
total of 55625 mesh points, while those with
meshes 3 and 4 used a 33 x 33 x 119 mesh
(129591 mesh points). In each case the mesh
was refined considerably near solid surfaces

and also near the leading and trailing edges.
For the bellmouth endwall shape a limited
investigation of mesh distribution was
performed by varying the refinement at the
leading and trailing edges for the same
overall number of mesh points. Results from
all the different mesh densities and distri-
butions are presented.

The convergence of the program is accelerated
using a multigrid algorithm. Previous
experience indicated that good convergence
was obtained after 2000 time steps and so
this number was used for the results presen-
ted here. At this point the rms value of
axial momentum residue was 2.6×10^{-4} on mesh
4. The program was run on a Stardent 1500
mini-supercomputer and 2000 time steps on
mesh 4 required approximately 43 hours of cpu
time.

5.   MESH COMPARISON

The four meshes employed for modelling the
bellmouth profile are compared in figures 3
and 4. The different mesh refinements at
leading and trailing edges and near solid
surfaces resulted in the mesh spacings shown
in table 1. The endwall boundary layer at
the first plane, one axial chord upstream of
the leading edge was set to have a thickness
of 6% annulus height. This resulted in 6
points in this inlet boundary layer for
meshes 1 and 2, and 8 points for meshes 3
and 4.

The mid-height distributions of isentropic
Mach number predicted with the different
meshes are compared in figure 5. The main
variations occur on the pressure surface
between the leading edge and 60% axial chord
(Cax) and on the suction surface at about 60%
Cax. The latter feature corresponds to the
impingement of the trailing edge shock on to
the suction surface and the differences are
probably due to the different modelling of
the trailing edge flow with the various
meshes. This shock is modelled best with
mesh 4, which has the finest spacing at the
trailing edge. Contours of Mach number near
the trailing edge are illustrated for meshes
2 and 4 in figure 6. The pressure surface
boundary layer upstream of the trailing edge
is considerably thinner with the finer mesh
leading to delayed separation and higher peak
Mach number at the trailing edge. This
produces the stronger shock evident where it
impinges on the suction surface. On the
forward pressure surface the isentropic Mach
number is higher with the finer meshes than
the coarser. The differences appear to be
associated with the finer cross-stream mesh
rather than the axial refinement, suggesting
that they are due to the improved modelling
of the pressure surface boundary layer as a
result of the finer spacing of meshes 3 and 4
in this region.

Figure 7 compares the secondary flows near
the suction surface for meshes 1 and 4. The
manner in which the endwall crossflow was
swept onto the suction surface can be clearly
seen in both cases. The finer mesh (4) has

resolved two separate points of impingement
onto the suction surface for each endwall at
approximately 50% Cax and 65% Cax with the
flow nearly parallel to the endwall between
the two. The coarser mesh has modelled the
first impingement very similarly to th. finer
mesh but has not picked up the second.

The development of mass-averaged loss along
the passage is shown for each mesh in figure
8. The results from the finer meshes have a
lower overall loss than those from the
coarser meshes reflecting the improved
modelling of the blade and endwall boundary
layers with the finer meshes. Also they
exhibit a more realistic growth of loss
through the blade passage; the loss with the
coarser meshes grows from the leading edge to
about 30% Cax, then diminishes before growing
again towards the trailing edge. At the
trailing edge mesh 4 shows a very rapid
growth of loss with little further growth
downstream whereas the other results,
particularly meshes 1 and 3, show less growth
at the trailing edge but significant further
growth downstream. This is attributed to the
increased mixing of the flow at the trailing
edge with mesh 4 due to the finer mesh
spacing in this region. The total losses at
the exit plane (1.0 axial chord downstream of
the trailing edge) for meshes 1 and 4 are
0.081 and 0.052 respectively.

Contours of total pressure on a cross-stream
plane at 120% axial chord are compared for
all four meshes in figure 9. The distinctive
curved wake is due to the curved trailing
edge and the radial variation of exit angle.
In each case the peak loss (minimum total
pressure) occurs in a thick region around
mid-height, the level of this peak loss being
similar for all the meshes. Meshes 1 and 4
give narrower wakes than (respectively)
meshes 2 and 3 suggesting that the finer mesh
in the trailing edge region has changed the
modelling of the flow in the mixing region
downstream of the trailing edge. Also, the
wake with mesh 4 is narrower than with mesh 1
indicating that the improved modelling of the
suction and pressure surface boundary layers
due to the finer near-wall mesh has resulted
in thinner boundary layers being predicted at
the trailing edge.

6. TURBULENCE MODEL COMPARISON

The above results were all produced using the
Baldwin-Lomax turbulence model with the
specified transition locations. Calculations
have also been performed using the one
equation turbulence model on meshes 1 and 4.
The development of the mass averaged loss
along the vane passage is compared for these
meshes with both turbulence models in figure
10. There are only small differences between
the results with the two turbulence models,
particularly with the coarser mesh. This is
consistent with the findings of Dawes (Ref 8)
which he attributes to most of the loss being
generated very close to solid surfaces,
within a laminar sub-layer region which is
common to both models.

7. COMPARISON WITH EXPERIMENT

The total pressure traces obtained from the
centre hole of the three hole probe for the
experimental sweeps at 10% and 50% vane
height are shown in figure 11. A small
amount of smoothing has been applied to the
traces to remove some high frequency noise.
The vane wakes can be seen clearly in both
plots, the wake at 50% height being consider-
ably wider than that at 10% height. Also
evident are some oscillations in the traces
between the wakes. These are due to fluctu-
ations with time in the total pressure up-
stream of the vanes arising from piston
velocity variations during the operation of
the rig. Because of the manner in which the
traverse is performed, a variation with time
appears as a variation with position when the
trace is plotted. These oscillations can be
largely removed by further analysis to allow
for the variation in upstream total pressure;
the results at 10% and 50% height after this
additional processing are shown in figure
12.

The individual experimental sweeps, after
processing, have been combined to form a
complete area traverse and contours of total
pressure are shown in figure 13 together with
the predictions using both turbulence models
on mesh 4. The predictions both show wakes
which are wider at mid-height than near the
endwalls. The wake in the prediction with
the one-equation model is wider (23% pitch at
mid-height) than that with the Baldwin-Lomax
model (20%); the experimental results show a
wake which is slightly wider than either
prediction (25%). The one-equation model
predicts a similar deficit of total pressure
in the centre of the wake to the experimental
value but the Baldwin-Lomax model does not
capture the full deficit. In common with the
experimental results neither prediction has
identifiable regions of secondary loss though
the Baldwin-Lomax results do exhibit some
thickening of the wake near the hub. The
experimental results show greater curvature
of the wake near mid-height and a more acute
angle between the wake and the endwall,
particularly near the tip, than either pre-
diction, with again the one-equation model
giving a slightly closer result.

8. COMPARISON OF ENDWALL PROFILES

Measurements of surface static pressure were
taken for both the bellmouth and S-bend
endwall profiles. These are shown on figure
14 in the form of isentropic Mach number
distributions at 10% and 50% span, together
with predictions for both endwall profiles,
using mesh 1 spacings. At 10% span the
S-bend hub profile has increased the Mach
number on the early part of the suction
surface and slightly increased the strength
of the shock at approximately 70% Cax. The
effect on the pressure surface is small. At
50% span there is very little difference
between the two sets of experimental results,
though the S-bend does still have a slightly
higher Mach number over the early suction

6

surface. Each of these differences has been
well modelled by the flow program, though
neither prediction captures the full shock
strength. The effect of the endwall profile
on the predicted secondary flows is illus-
trated in figure 15 by the velocity vectors
near to the suction surface. It appears that
the passage cross-flow on the hub impinges on
to the suction surface slightly further down-
stream with the S-bend profile but that it
does so at a greater angle resulting in a
greater spanwise extent of the vane being
affected by the secondary flow. The effect
on the losses is small as is shown by
contours of total pressure on the plane at
120% Cax in figure 16. There is very little
difference between the shape of the vane
wakes or the total pressure deficits for
either endwall profile. The total loss is
0.081 of exit dynamic head for the bellmouth
profile and 0.084 for the S-bend.

9.   CONCLUSIONS

A transonic turbine nozzle guide vane has
been tested in an annular cascade facility
with two different endwall profiles. Vane
surface static pressures have been measured
and a traverse of the total pressure field
downstream of the trailing edge has been
performed for one of the profiles.

The new area traversing technique has given
good results. It has been found necessary to
process the results to remove the effect of
piston oscillation.

The Dawes three-dimensional viscous flow
program has been used to model the flow
through the nozzle. A variety of mesh densi-
ties and distributions have been investigated
as have two different turbulence models.
Good agreement has been obtained for the
pressure distributions and realistic second-
ary flows have been predicted. It was found
that the finer meshes gave better predictions
of the static pressure distributions and
reduced levels of loss as a result of the
improved modelling of the vane surface
boundary layers.

The different turbulence models were found to
have little effect on the predicted overall
loss though there were differences in the
distribution and the shape of the vane wakes.
The one-equation turbulence model produced
wakes which had similar levels of total
pressure deficit to the experiment and had a
more similar shape than those with the
Baldwin-Lomax algebraic model.

REFERENCES

1.   R. C. Kingcombe, J. D. Bryce and
N. P. Leversuch, "Design and Test of a High
Blade Speed, High Work Capacity Transonic
Turbine", AGARD CP-421, 1987.

2.   R. C. Kingcombe, S. P. Harasgama,
N. P. Leversuch and E. T. Wedlake, "Aero-
dynamic and Heat Transfer Measurements on
Blading for a High Rim-Speed Transonic

Turbine", ASME Paper 89-GT-223.

3.   A. J. Brooks, D. E. Colbourne,
E. T. Wedlake, T. V. Jones, M. L. G. Oldfield
D. L. Schultz and P. J. Loftus, "The
Isentropic Light Piston Annular Cascade
Facility at RAE Pyestock", AGARD CP-390, 1985

4.   G. C. Horton, "Secondary Flow Predic-
tions for a Transonic Nozzle Guide Vane",
AGARD CP-469, 1990.

5.   W. N. Dawes, "A Numerical Method for
the Analysis of 3D Compressible Flow in
Turbine Cascades; Application to Secondary
Flow Development in a Cascade With and With-
out Dihedral", ASME Paper 86-GT-145, 1986.

6.   B. S. Baldwin and H. Lomax, "Thin
Layer Approximation and Algebraic Model for
Separated Turbulent Flows", AIAA Paper 78-257
1978.

7.   N. T. Birch, "Navier-Stokes Predic-
tions of Transition, Loss and Heat Transfer
in a Turbine Cascade", ASME Paper 87-GT-22,
1987.

8.   W. N. Dawes, "A Comparison of Zero and
One Equation Turbulence Modelling for Turbo-
machinery calculations", ASME Paper 90-GT-303
1990.

Fig 1

	Mesh 1	Mesh 2	Mesh 3	Mesh 4
Axial spacing at leading edge (% axial chord)	0.36	0.43	0.30	0.10
Axial spacing at trailing edge (% axial chord)	0.36	0.43	0.30	0.10
Spanwise spacing at endwall (% blade height)	0.39	0.49	0.24	0.24
Tangential spacing at blade surfaces (% pitch)	0.39	0.39	0.24	0.24

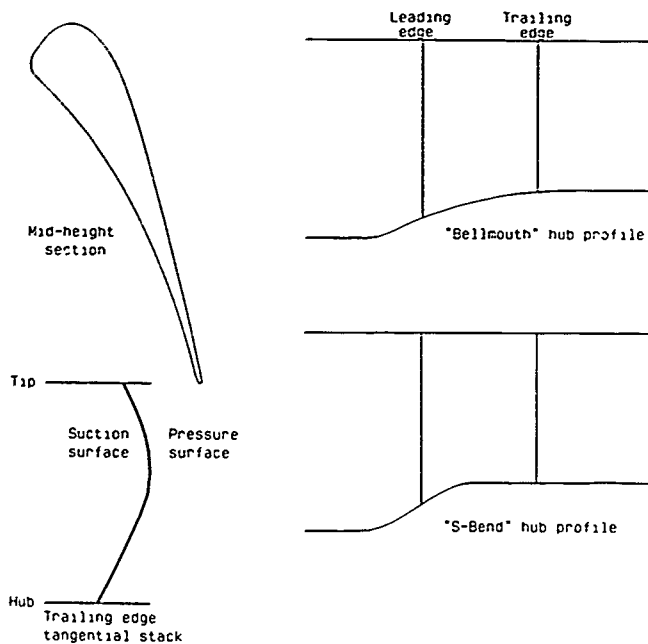Table 1. Mesh spacings for different meshes

Mid-height
section

Leading edge   Trailing edge

"Bellmouth" hub profile

Tip

Suction surface   Pressure surface

"S-Bend" hub profile

Hub
Trailing edge
tangential stack

Fig 1   Illustration of ngv design

straight tip

swan-necked tip
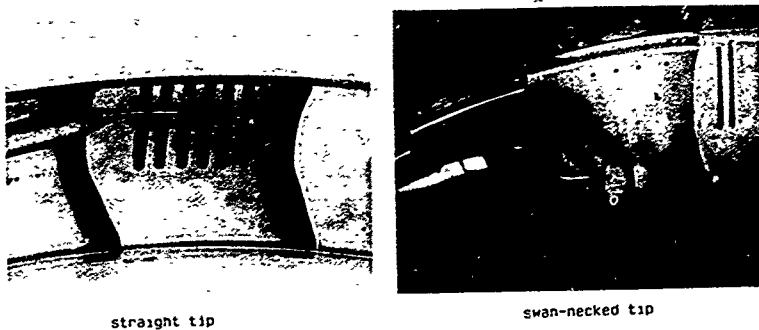
Fig 2. The two traverse probes

Mesh 1
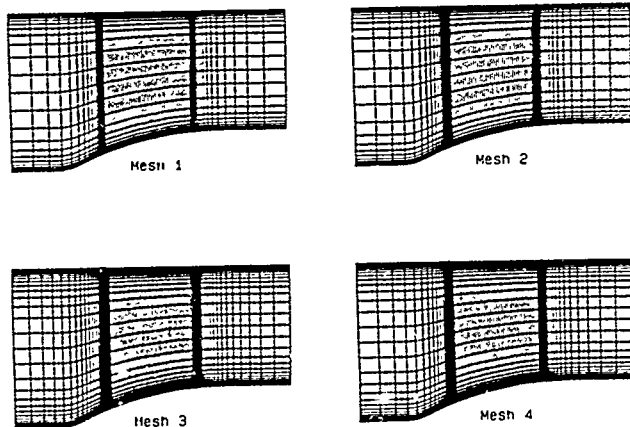
Mesh 2

Mesh 3

Mesh 4

Fig 3 Details of calculation mesh

Fig 4
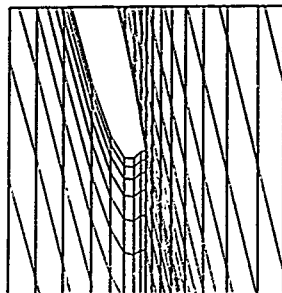
Mesh 1
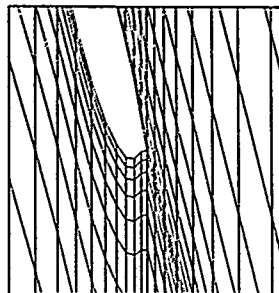
Mesh 2
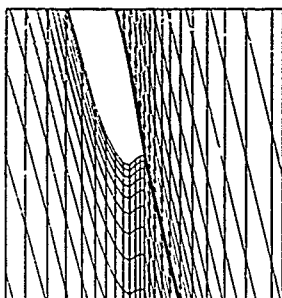
Mesh 3
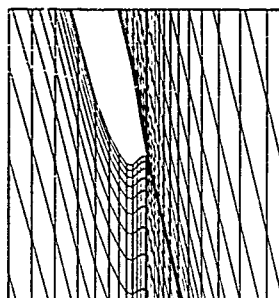
Mesh 4

Fig 4 Details of mesh near trailing edge
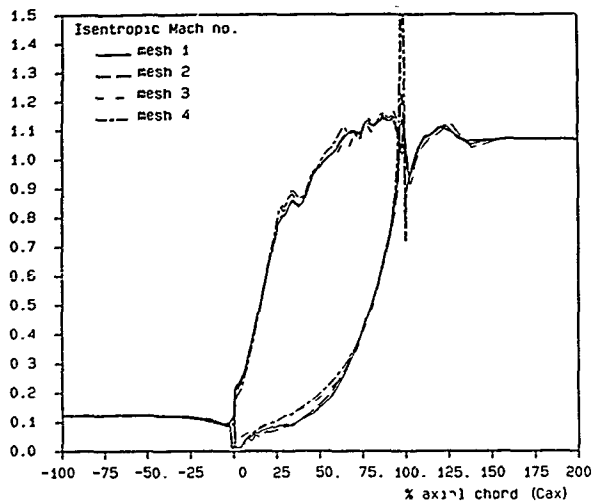
Fig 5   Comparison of mid-height isentropic Mach number

distributions with different meshes
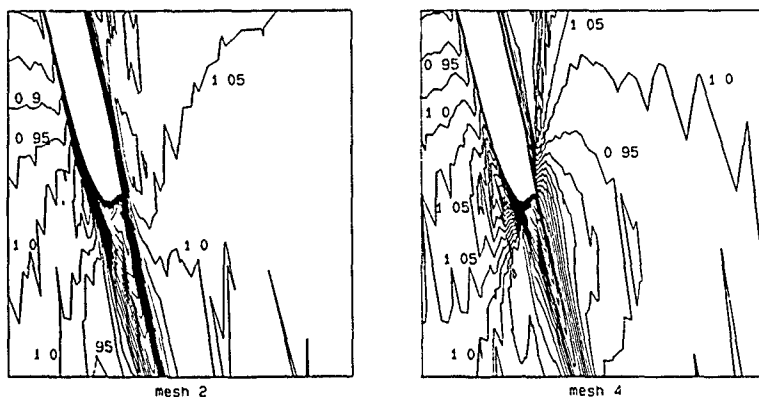
Fig 6   Contours of Mach number near trailing edge
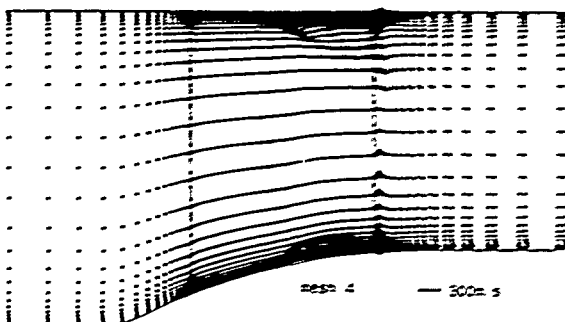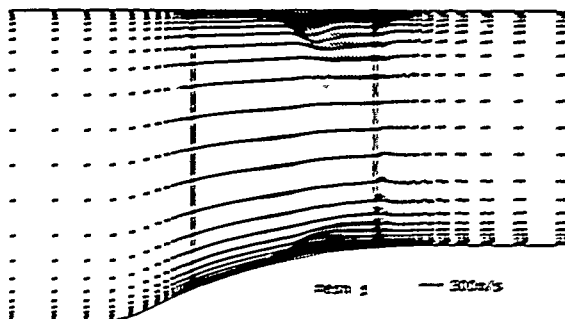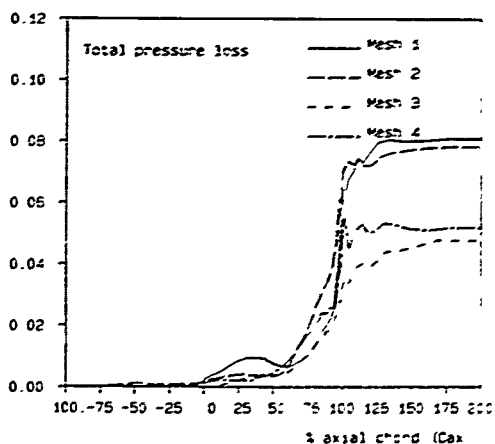
Fig 7. Suction surface flows with meshes 1 and 4

Fig 8 Development of total pressure loss with different meshes

Fig9
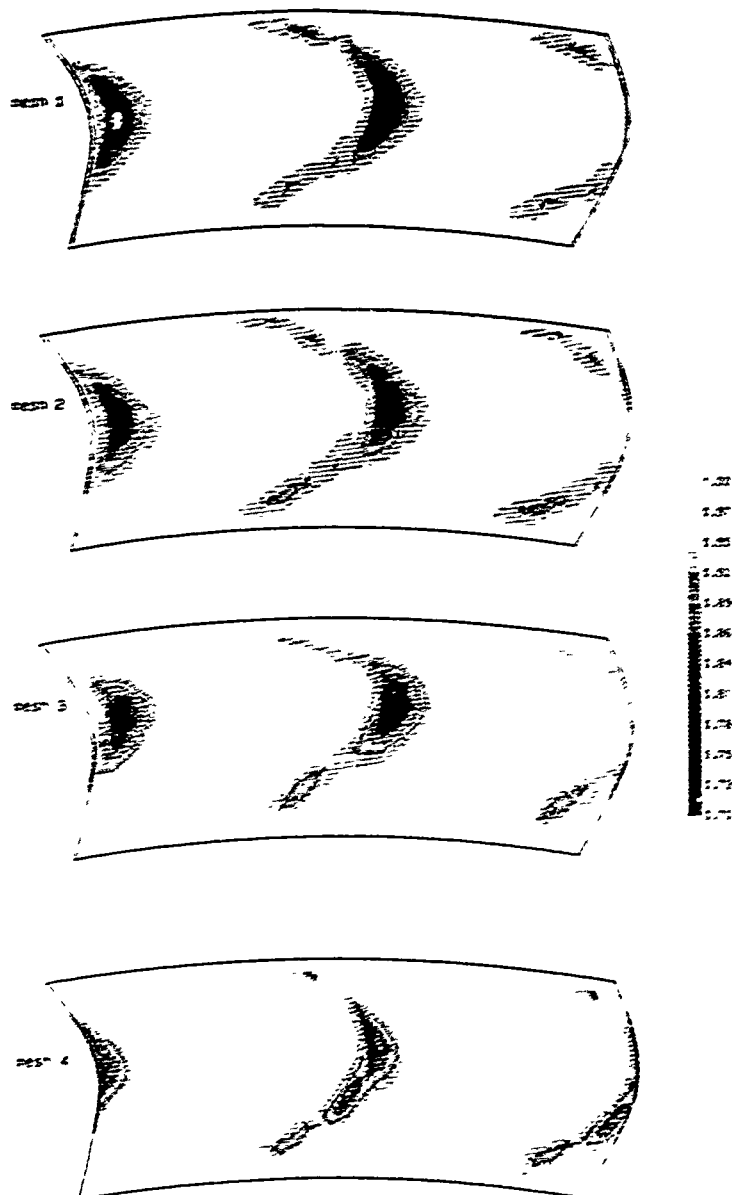
Fig 9 Contours of total pressure on a plane at 120% axial chord
Comparison of results with different meshes

Fig 10 11 12
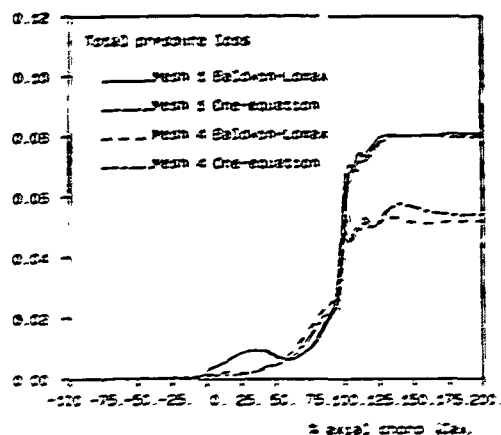
Fig 10. Development of total pressure loss with different turbulence models
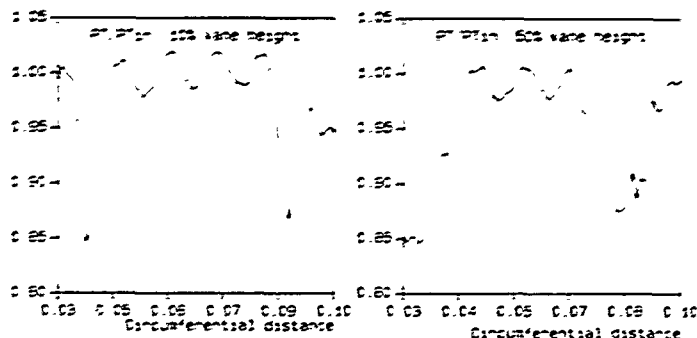
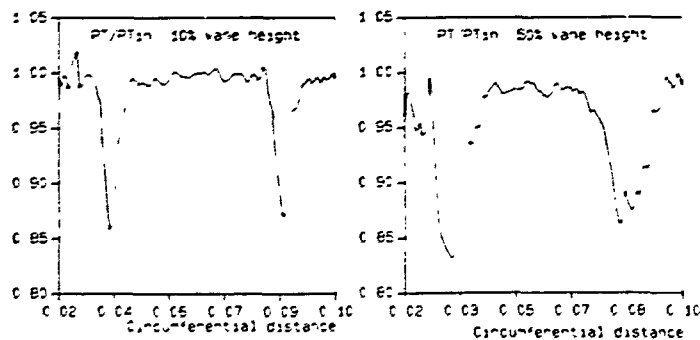Fig 11. Total pressure traverse sweeps before processing

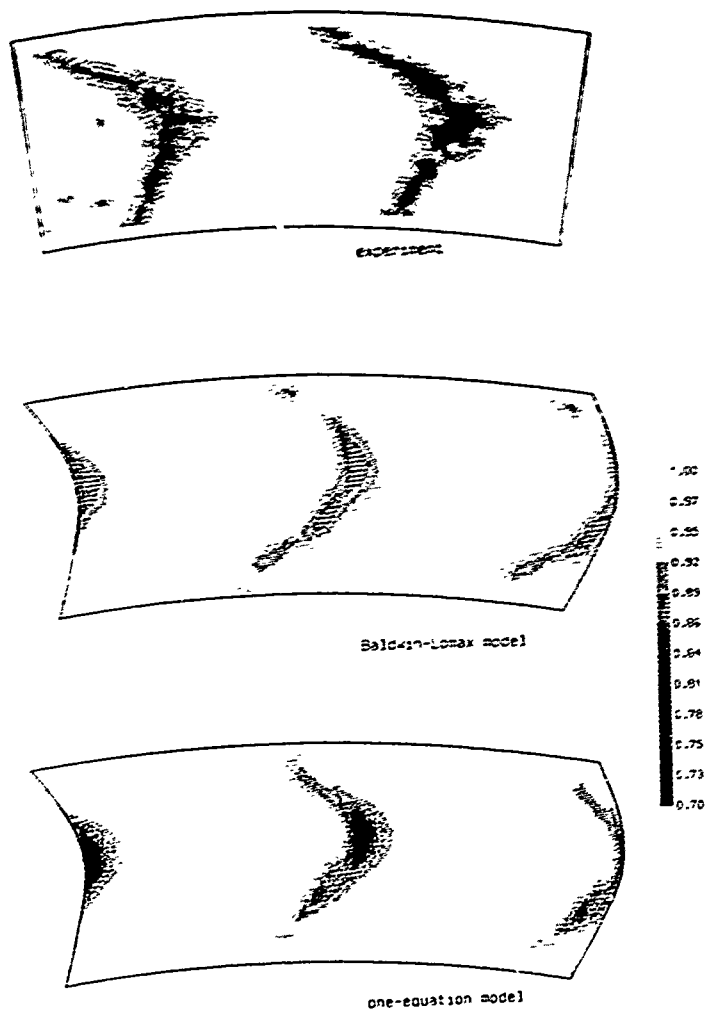Fig 12 Total pressure traverse sweeps after processing

Fig 13

Fig 13 Total pressure contours on a plane at 120% axial chord
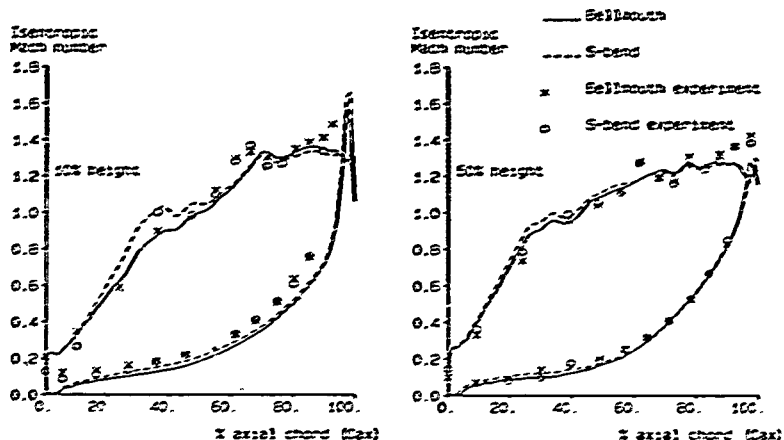Comparison of different turbulence models with experiment

Fig 14. Mach number distributions with different endwall profiles

bellmouth hub
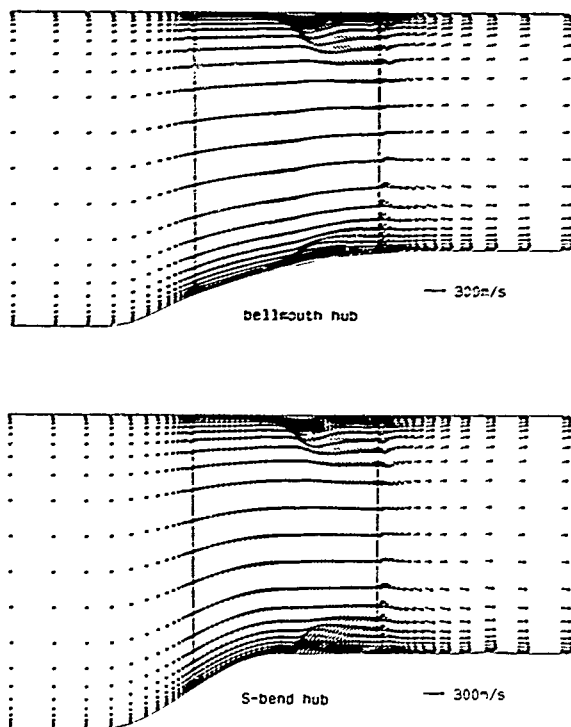

S-bend hub   — 300m/s

Fig 15 Suction surface flows with bellmouth and S-bend hub profiles

Fig 16

bellmouth hub
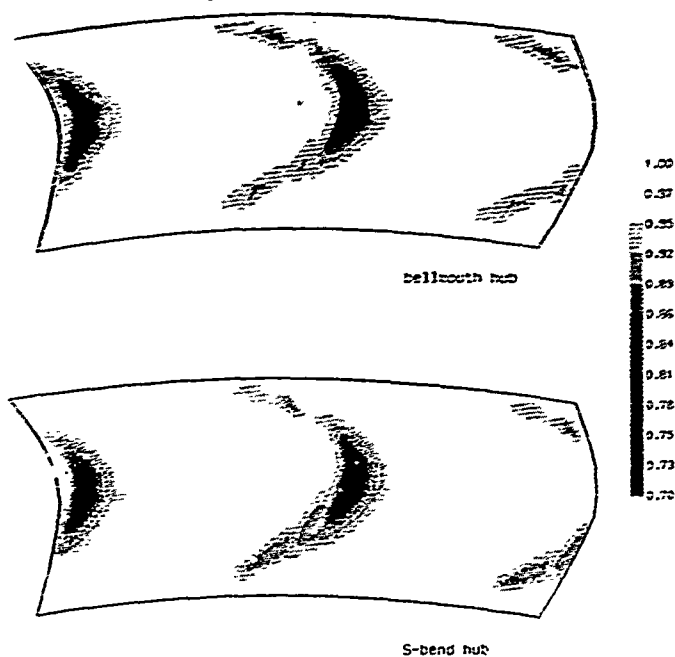
S-bend hub

Fig 16   Total pressure contours on a plane at 120% axial chord
for bellmouth and S-bend hub profiles

1. DRIC Reference (to be added by DRIC)	2. Originator's Reference RAE TM P 1205	3. Agency Reference	4. Report Security Classification/Marking UNLIMITED

5. DRIC Code for Originator 7674200W	6. Originator (Corporate Author) Name and Location Royal Aerospace Establishment, Pyestock, Hants, TE

5a. Sponsoring Agency's Code	6a. Sponsoring Agency (Contract Authority) Name and Location

7. Title Predictions and measurements of 3D viscous flow in a transonic turbine nozzle guide vane row

7a. (For Translations) Title in Foreign Language

7b. (For Conference Papers) Title, Place and Date of Conference

8. Author 1. Surname, Initials Horton, G.C.	9a. Author 2 Harasgama, S.P.	9b. Authors 3, 4 ... Chana, K.S.	10. Date April 1991	Pages 21	Refs. 8

11. Contract Number	12. Period	13. Project	14. Other Reference Nos.

15. Distribution statement
    (a) Controlled by —

    (b) Special limitations (if any) —

If it is intended that a copy of this document shall be released overseas refer to RAE Leaflet No.3 to Supplement 6 of MOD Manual 4.

16. Descriptors (Keywords)   (Descriptors marked * are selected from TEST)
Turbines*. Viscous flow.

17. Abstract
In recent years improvements in algorithms and computing power have allowed the regular use of three-dimensional viscous flow programs to analyse the flows in turbines. Before these programs can be used with confidence by the turbine designer, however, they must be validated by comparison with high quality experimental data taken at realistic conditions.

A transonic turbine nozzle guide vane has been tested in an annular cascade with two different endwall geometries. The measurements were taken at engine-representative flow conditions and include surface static pressures and a down-stream area traverse of total pressure.

The flow through these geometries has been modelled at the test conditions using a three-dimensional viscous flow program. The effects of different mesh densities and two turbulence models have been studied. Predictions of secondary flow and loss have been obtained and are compared with the experimental measurements.

RAE Form A143 (revised October 1980)

END
FILMED

DATE: 10-91

DTIC